

Chapter 17

Creating a 3D Assembly Drawing

In this chapter, you will learn the following to World Class standards:

- 1. Making your first 3D Assembly Drawing**
- 2. The XREF command**
- 3. Making and Saving Part One, the Rail**
- 4. Making and Saving Part Two, the Leg**
- 5. Making and Saving Part Three, the Brace**
- 6. Making and Saving Part Four, the Foot Rest**
- 7. Creating the Assembly File**
- 8. Using the XREF Tool to Import the First Part Drawing**
- 9. Using the XREF Tool to Import the Second Part Drawing**
- 10. Using the XREF Tool to Import the Third Part Drawing**
- 11. Using the XREF Tool to Import the Fourth Part Drawing**

Making Your First Assembly Drawing

In the world of three dimensional computer aided design, you can generate a group of solid parts and bring them together into a collection of joining components to create a single product. This newer system of design is replacing the older and antiquated two-dimensional layout. The fresher arrangement of virtual apparatus allows the designer to examine interfaces between parts and look for interferences. With the ability to add material and lighting, you can inspect the assembly for cosmetic appearance as well. If developing solid parts in the earlier chapters' added inspiration to your professional character and generating multiple orthographic views in seconds gave you a sense of modern day efficiency, that's good; for any true designer, the ability to create an inexpensive and complex assembly is breathtaking. An architect or engineer who has control of the 3D assembly technology through use of a computer can now feel they are in the driver's seat in the design process.

In the history of assembly drawings, such as those originated by Leonardo Da Vinci, hand-drawn artistic renderings were drawn to portray assemblies such as a church or defensive structures. Closer to the modern day, in the 20th century, layout drawings would show each part to half or full scale to identify the connections between objects. Once two-dimensional computer aided design systems were implemented, many of the similar processes used in the previous centuries proved adequate, until three-dimensional modeling arrived on the scene.

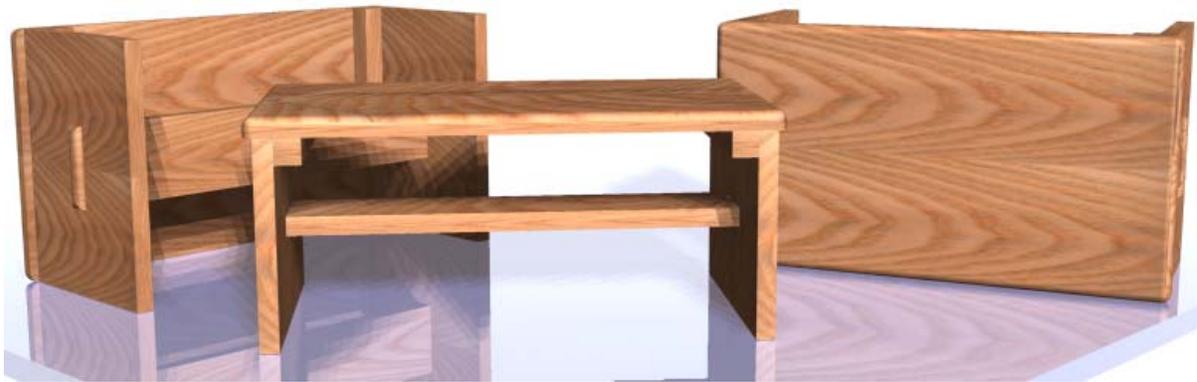


Figure 17.1 – A Marketing Image of your First Simple Three Dimensional Assembly

Able to create a straightforward compilation of wooden pieces like those shown in Figure 17.1, the three-dimensional assembly procedure will allow you to draw each part separately. Then in the assembly drawing file, the External Reference function can be used to assemble the solids into the final product. You are still able to build an orthographic layout in Paper Space, so overall dimensions, bill of material, and assembly notes can be added to explain to the technician how to erect the manufactured goods. In this lesson, you will obviously begin with a simple assembly that only has four parts. All are relatively trouble-free to construct in Model Space.

The XREF Command

The difference between a drawing reference and a drawing block is very important when beginning to build assemblies. Imagine a house with thousands of components. If you insert each part using “copy and paste” or the insert block function, the solid will become part of the home assembly with all the part’s definitions that existed in its individual part drawing file. Adding many parts in this way can bog down your computer’s speed because of the increasing file size. This can seriously affect your efficiency as a drafter.



The External Reference function solves the dilemma of large drawing file sizes. Instead of inserting an external block, the tool allows you to insert the reference material of the data into the assembly drawing. This process is akin to a modern web page: there are links between the home page and the other web pages in the server. progeCAD Professional creates a link to the part’s computer file and the assembly drawing does not absorb its entire set of data. You will need to add the Reference Toolbar as shown in Figure 17.2 to your graphical access many of the tools necessary to creating a reference assembly.

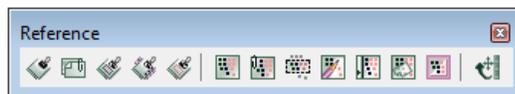


Figure 17.2 – Reference Toolbar

Making and Saving Part One, the Rail

As in the practice solids problems, the engineer or designer will provide you with a sketch. In the future, you can be the imaginative one who creates such illustrations to be formed into an assembly. In your first assembly drawing, the first solid part is just a rectangular box. The dimensions of the geometric solid are 1.0 inch by 1.0 inch by 11.0 inches as shown in Figure 17.3. Again, the basic tools like the Box command will come in handy to create simple solids like this.

As a learning point, when many junior designers receive a set of sketches from their supervisor, they will pick the most difficult geometric shape to load into model space first. In a perfect world where the customer or production is not under deadline, this technique would be satisfactory because you would be delivering all of the drawings at once in a complete package. For this to work your manager would have to be running a serial production system where each design step runs to completion before phasing into the building of the prototype. However, many supervisors use a parallel method to speed through the design process. In this case, you would select the easiest part or a component with a long lead or manufacturing time to draw first. As you finished the virtual prototype in Model Space the first production unit would be built simultaneously on the shop floor.

This design does not have any lengthy deliverable items, so you can start with the easiest part

first. Follow the 3D modeling procedures learned from your earlier work. Always refer to the sketch as well as the written procedures. Eventually in your career, the sketch will be your only guide to drawing the part.

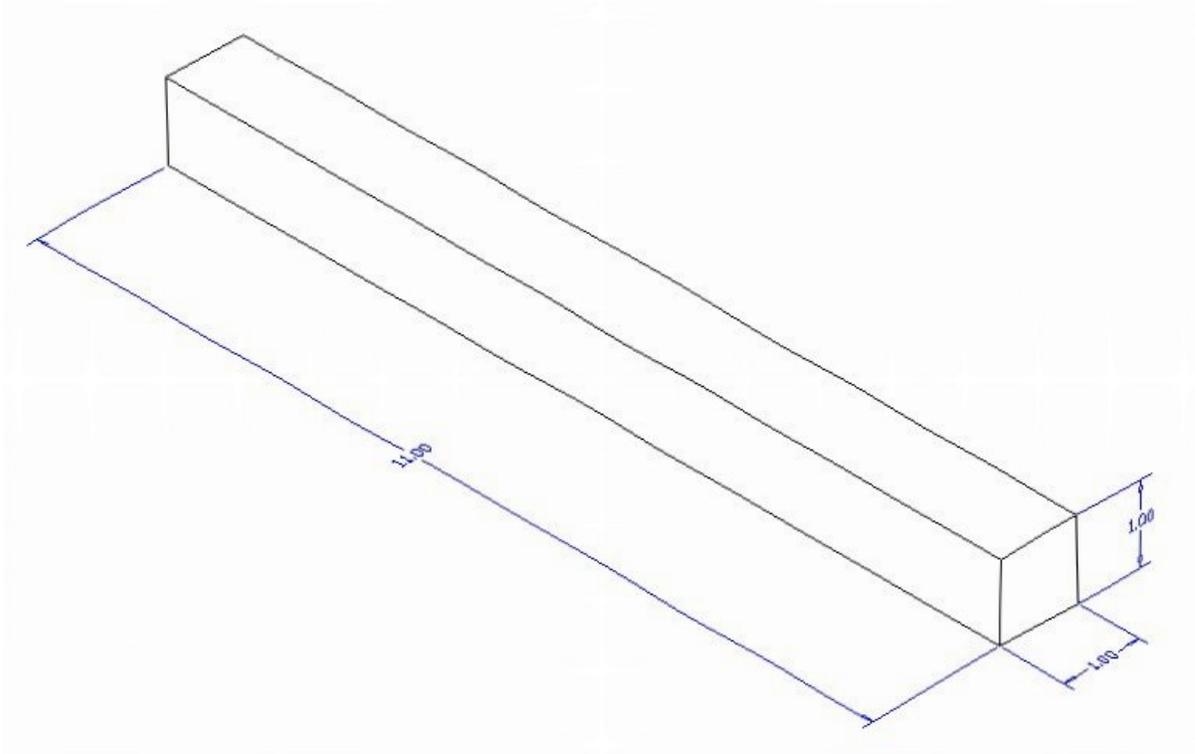
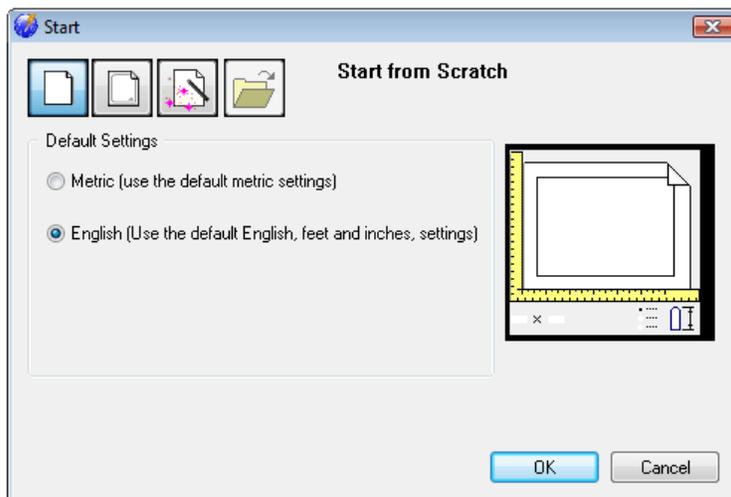


Figure 17.3 – Part One Sketch

Starting a the first 3D Part Drawing, the Rail



In Figure 17.4, the “Start” window is loaded by selecting the New tool on the Standard Toolbar. Select the “Start from Scratch” button at the top of the window. Select English as the default units setting and hit OK.

Figure 17.4 – Starting the Drawing from Scratch

To draw the solid rectangular box, select the Box tool on the Solids toolbar. The command line prompts you to specify the corner or center of the box. Identify the corner of the box by picking anywhere in the middle of the graphical display. After picking the starting point for the box, again the system asks you to denote a corner; type **L** for length and hit **ENTER**. For this part, type **1.0** for the X coordinate, **11.0** for the y coordinate and **1.0** for the Z coordinate.

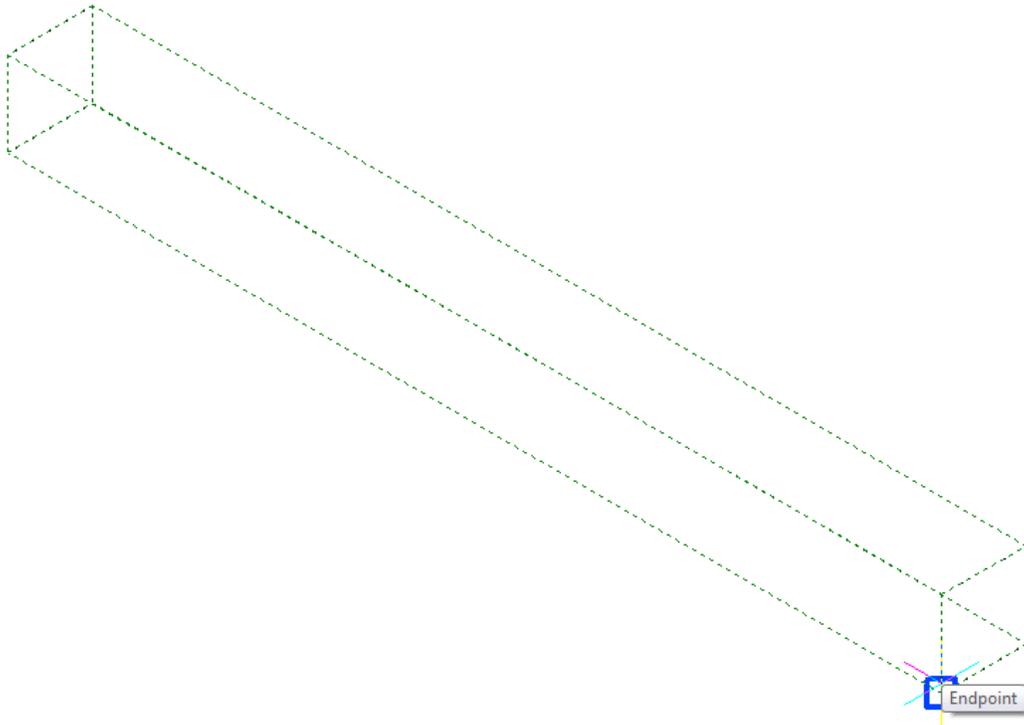


Figure 17.5 – Creating Part One and Moving the Part to the Origin

Select the Southwest Isometric view to better view the box. To move part one to the drawing origin of 0,0,0 on the x, y and z axis, pick the Move tool on the Modify toolbar, select the 3D solid, and hit ENTER. At the prompt to specify a base point, select the endpoint shown in Figure 17.5 at the lower corner of the edge closest to you, which will become the part's insertion point into the assembly drawing. At the “displacement point” prompt, type **0,0,0** and hit **ENTER**.

To save Part 1, select the Save tool on the Standard toolbar. The Save Drawing As window will appear in your graphical display. In the Save In list box, select your drawing folder. You may want to create a new folder specifically to store the part references drawings from this chapter. Select the new folder tool, which is the yellow folder with a sparkle in the upper right corner as shown in Figure 17.6. When the folder with the name “new folder” appears in the Save Drawing As display, type the words “Foot Rest Assembly” and hit ENTER. Double click on your new assembly folder to open the Foot Rest Assembly folder. In the File Name textbox, type “Rail” and click the Save button to save the drawing (Figure 17.7).

You can finish many simple parts in every assembly in minutes. Using smart procedures like these can significantly increase your productivity.

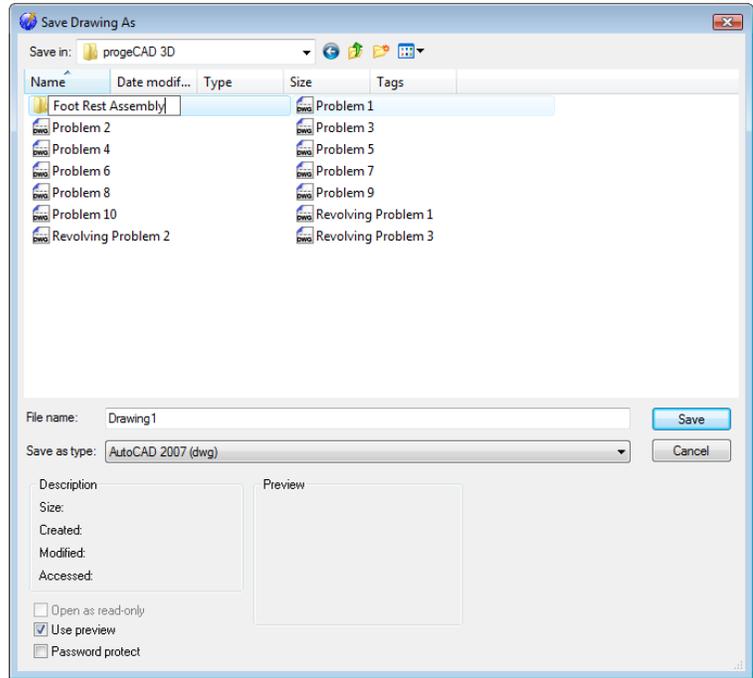


Figure 17.6 – Creating a Folder for the Foot Rest Assembly Drawing

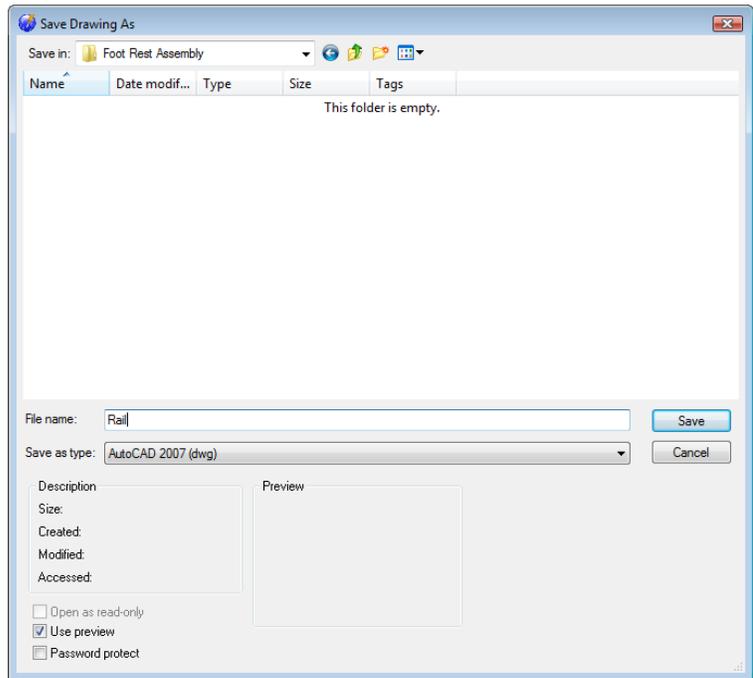


Figure 17.7 – Saving the Rail File in the Foot Rest Assembly Folder

Making and Saving Part Two, the Leg

Part two has a more complex shape from the side view, so you will use the Polyline function to draw the perimeter and then extrude the shape. Start the problem from scratch like you did for the rail.

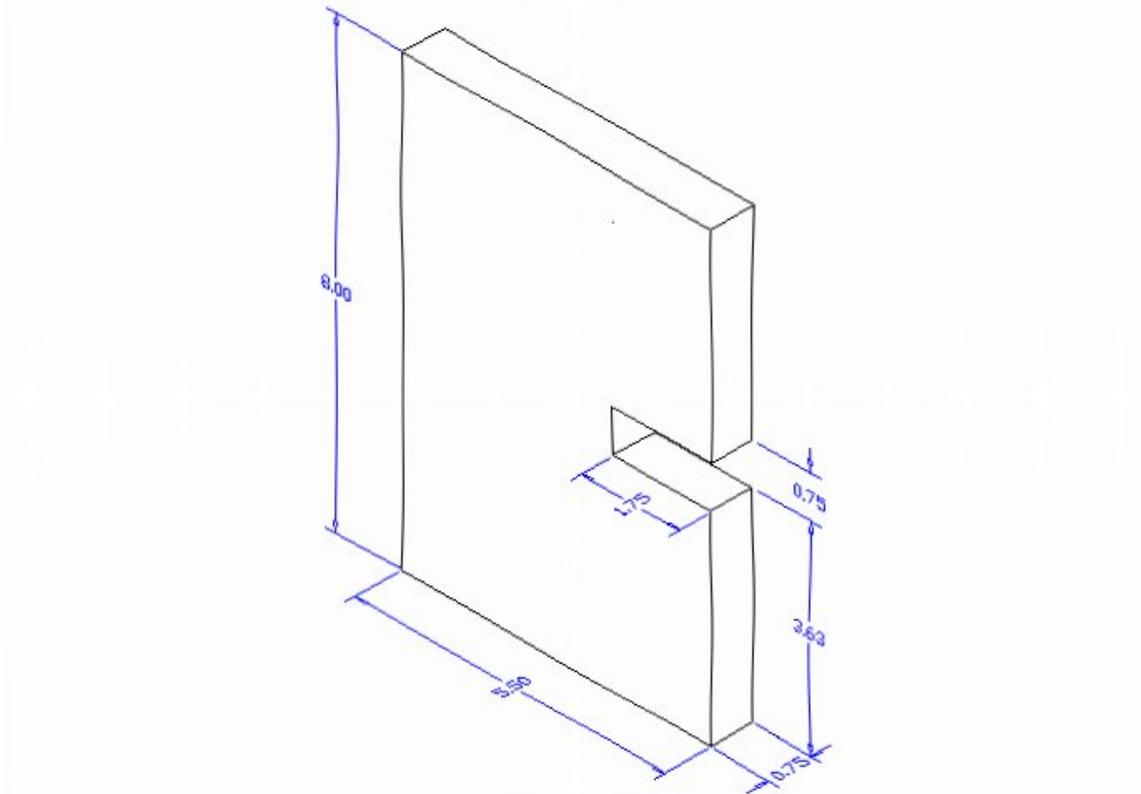
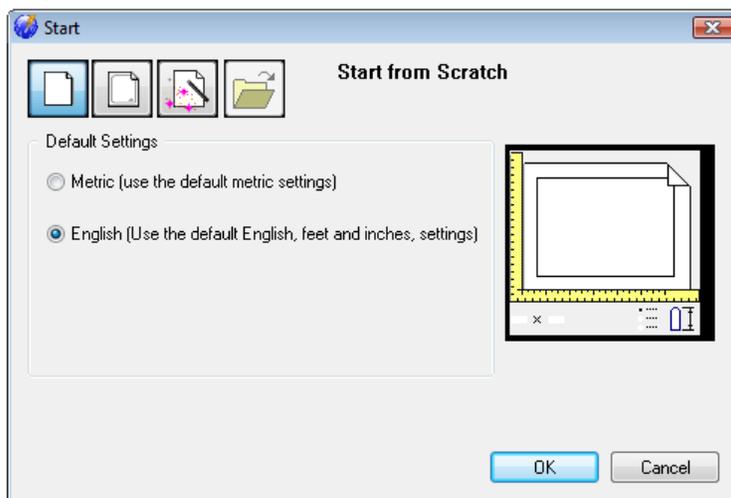


Figure 17.8 – Part Two Sketch



In Figure 17.9, the “Start” window is loaded by selecting the New tool on the Standard Toolbar. Select the “Start from Scratch” button at the top of the window. Select English as the default units setting and hit OK.



Figure 17.9 – Starting the Drawing from Scratch

Drawing a closed Polyline should not be new to you. You will have to draw the contour of this solid similarly to those drawn in previous chapters. On the Draw Toolbar, select the Polyline tool and specify a start point in the lower left hand corner of the graphical display.

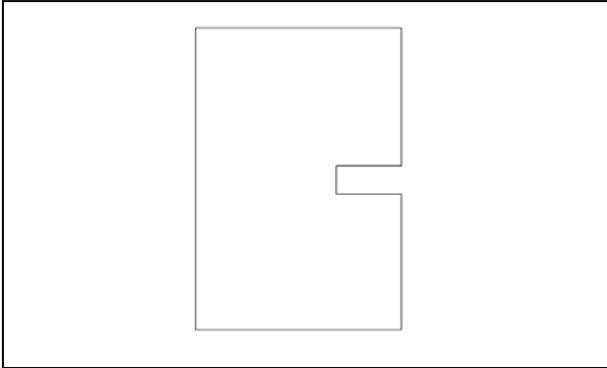


Figure 17.10 – Drawing the Shape

With the Ortho mode “on”, draw a **5.5** unit line to the right, a **3.625** line up, a **1.75** line to the left, a **0.75** line upward, a **1.75** line to the right, a **3.625** line upward, a **5.5** to the left and then type “**C**” to close the Polyline. A closed entity will appear in the graphical display as shown in Figure 17.10. Select the Southwest Isometric view before proceeding to the next step in the drawing process.

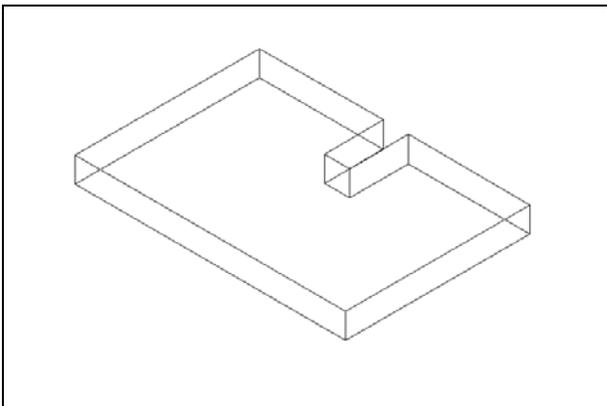


Figure 17.11 – Extruding the Shape

Select the Extrude tool on the Solids toolbar and the command line will prompt you to “**Select entities:**”. Pick the Closed Polyline anywhere on the outline and the command line will return with “**Entities in set: 1**”. Hit **ENTER** to specify the height or path of the solid. Type “**0.75**” for the height of the extrusion and for the angle of taper hit **ENTER** to accept the zero degree default. The shape will extend 0.75 units into the z axis as shown in Figure 17.11.

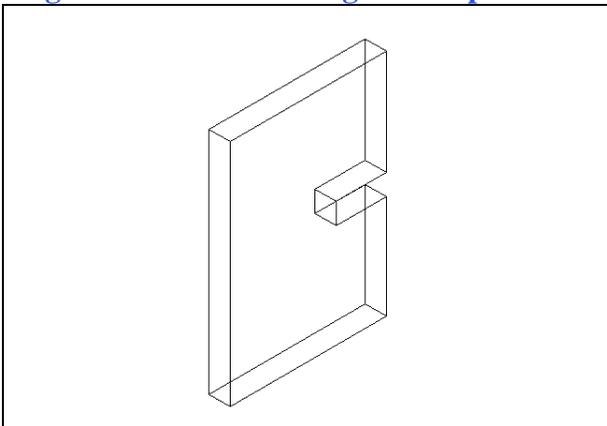
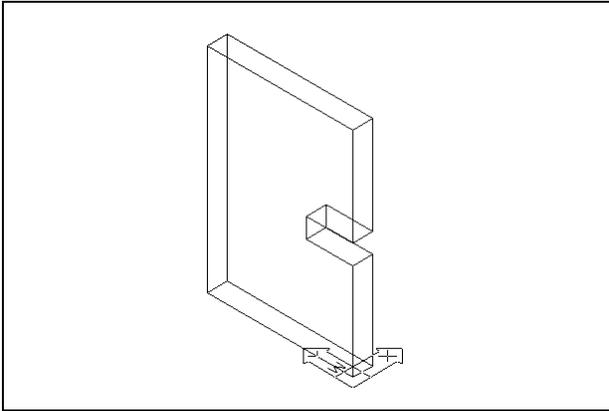


Figure 17.12 – Rotate3D the Solid

Select Modify on the Menu Bar, then 3D Operations and pick Rotate 3D from the list of commands. At the prompt, “**Select entities to rotate:**”, pick the solid and hit **ENTER** to proceed to the second part of the function. You are going to rotate the solid part on the x-axis, so pick any point on the display and then with the Ortho mode still “On” pick a second point to draw a line on the x-axis. Type **90** and **ENTER** to rotate the solid as shown in Figure 17.12.



To rotate the solid in the XY plane, select the Rotate command on the Modify toolbar and pick the solid. To continue with the command, hit **ENTER** to select the base point of rotation. Select any point at the base of the solid and then type **-90** for the degrees of counter clockwise rotation to finish this detail. Finally move the part to the origin using the lower corner closest to you as the base point (Figure 17.13). Now you are ready to save the part.

Figure 17.13 – Rotate the Solid

To save Problem Two in your Foot Rest Assembly folder, select the Save tool on the Standard toolbar. The Save Drawing As window will appear in your graphical display. In the Save In list box, select your drawing folder. At the File Name textbox, type “Leg” and hit the Save button to save the drawing (Figure 17.14).

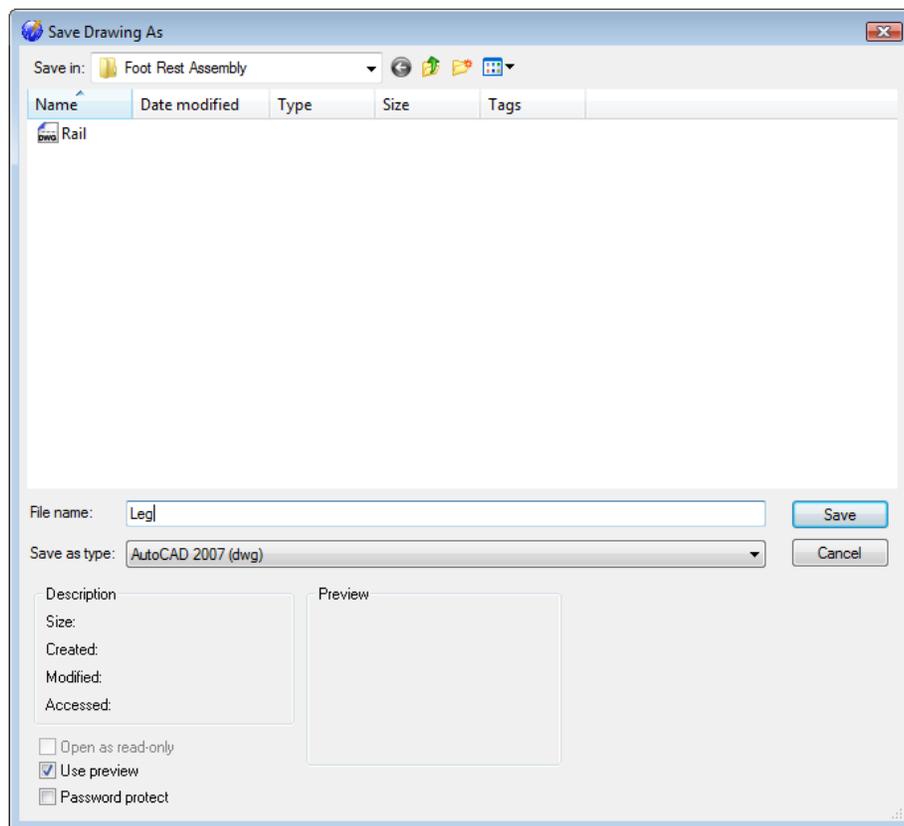


Figure 17.14 – Saving the Leg File in the Foot Rest Assembly Folder

Making and Saving Part Three, the Brace

Part three is a rectangular box with rounded ends. The dimensions of the geometric solid are 19.875 inches by 3.5 inches by 0.75 inches as shown in Figure 17.15. Again, the basic tools like the Box command will come in handy to create simple solids like this. You will use the Fillet command to round the corners of each end.

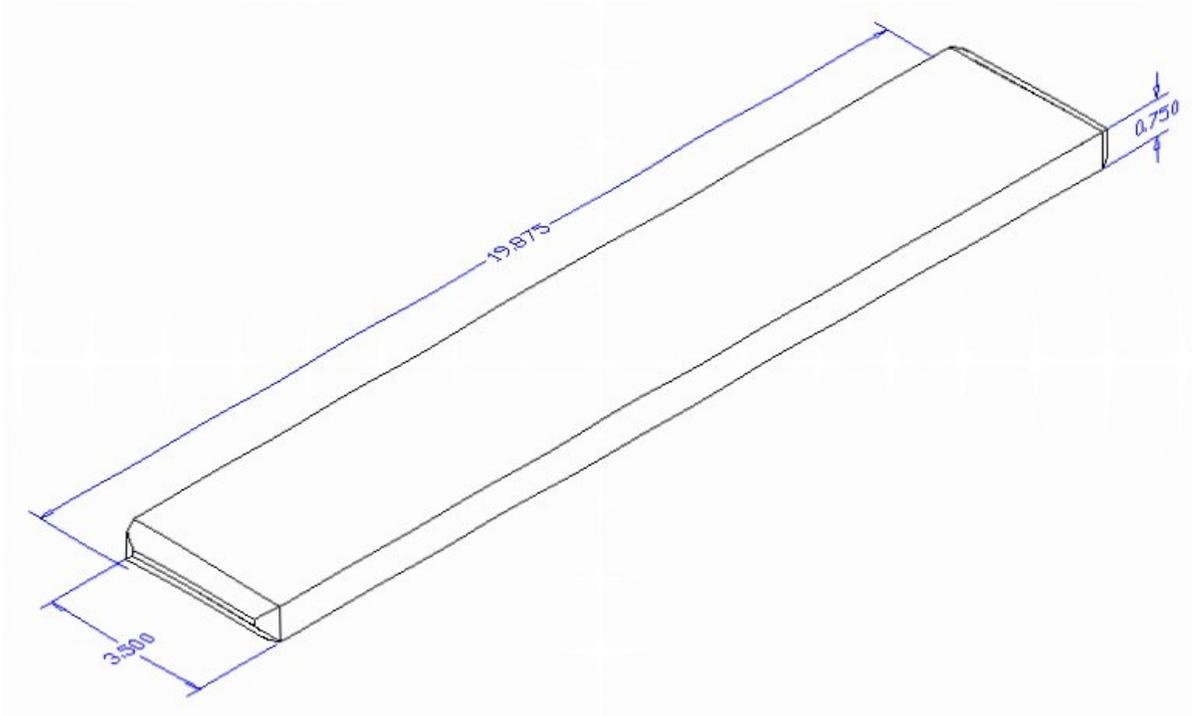
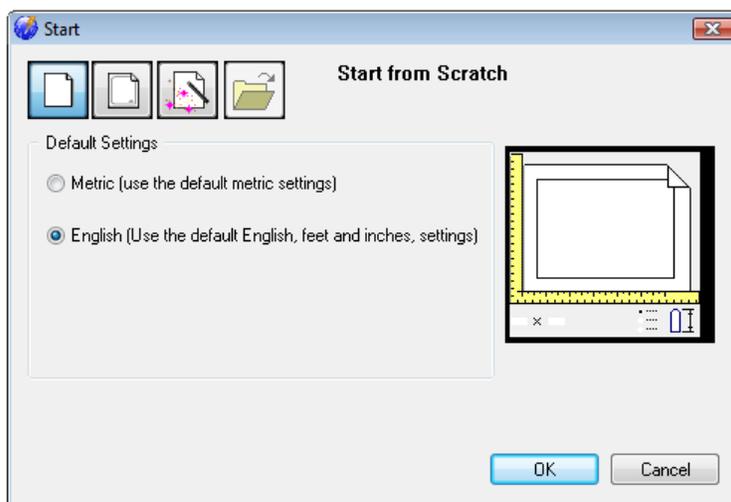


Figure 17.15 – Part Three Sketch



In Figure 17.16, the “Start” window is loaded by selecting the New tool on the Standard Toolbar. Select the “Start from Scratch” button at the top of the window. Select English as the default units setting and hit OK.



Figure 17.16 – Starting the Drawing Using the Mechanical Template

To draw the solid rectangular box, select the Box tool on the Solids toolbar. The command line prompts you to specify the corner or center of the box. Identify the corner of the box by picking anywhere in the middle of the graphical display. After picking the starting point for the box, again the system asks you to denote a corner; type **L** for length and hit **ENTER**. For this part, type **19.875** for the X coordinate, **3.5** for the y coordinate and **0.75** for the Z coordinate. Switch to the Southwest Isometric view so the solid appears in the graphical display as shown in Figure 17.17.

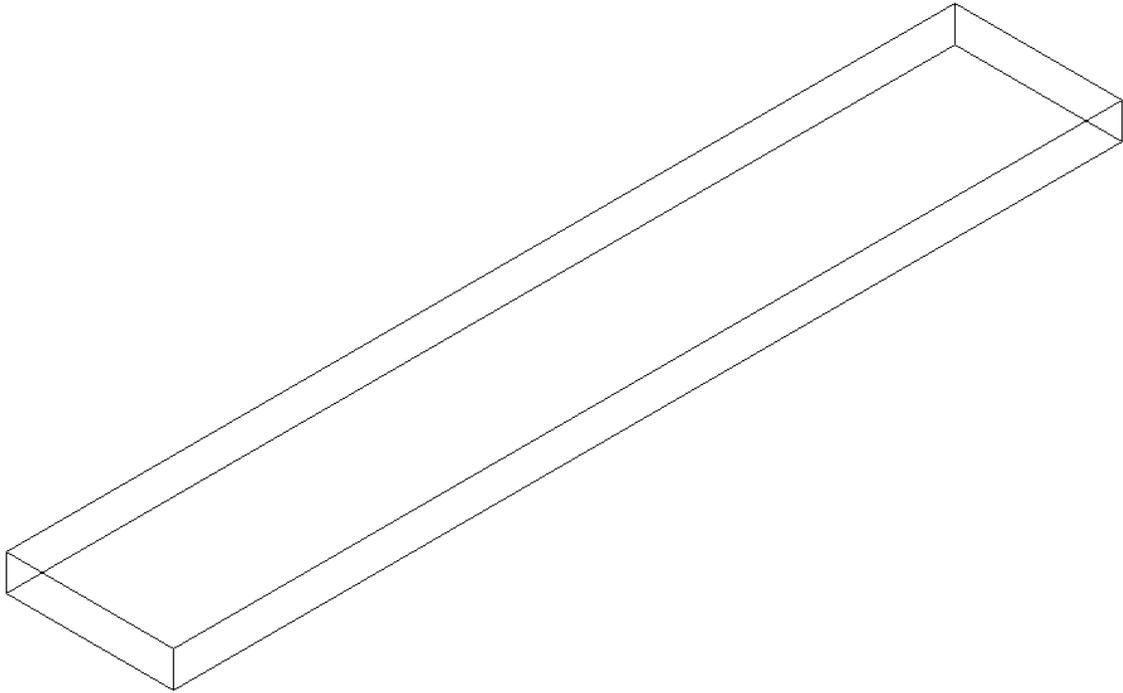


Figure 17.17 – The Solid Box

To move the part to the drawing origin of 0,0,0 on the x, y and z axis, pick the Move tool on the Modify toolbar, select the 3D solid, and hit ENTER. At the prompt to specify a base point, select the bottom corner closest to you, which will become the part's insertion point into the assembly drawing. Type **0,0,0** and hit **ENTER**.

Select the Fillet tool on the Modify toolbar. Type an “**R**” for radius and then **0.313** for the measurement. Select an edge on the left face of the box and hit ENTER. Next, select each edge of that face so that each line turns dotted and hit ENTER (Figure 17.18). The rounded edges will appear as shown in Figure 17.19. Repeat the process for the other side of part three and you will have symmetrical fillets on both ends (Figure 17.20).

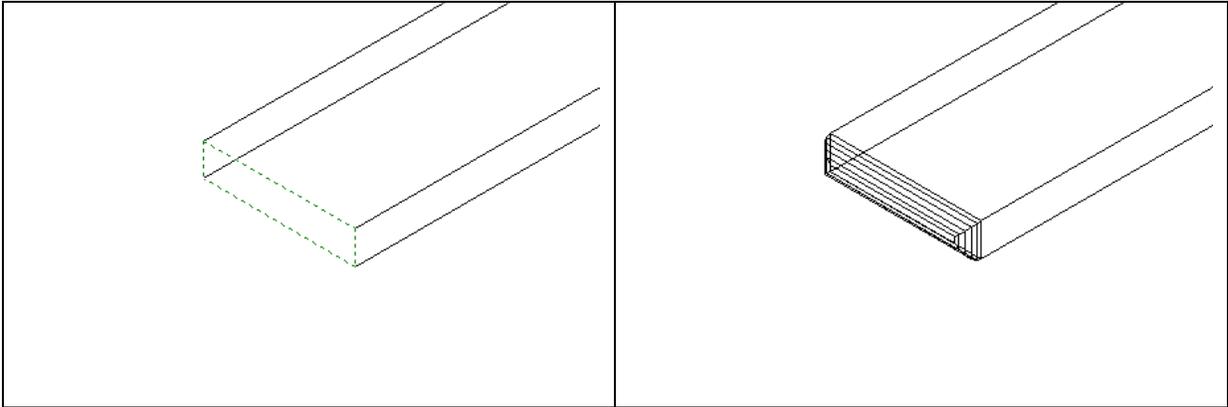


Figure 17.18 – Selecting the Edge

Figure 17.19 – The 0.313 Fillet

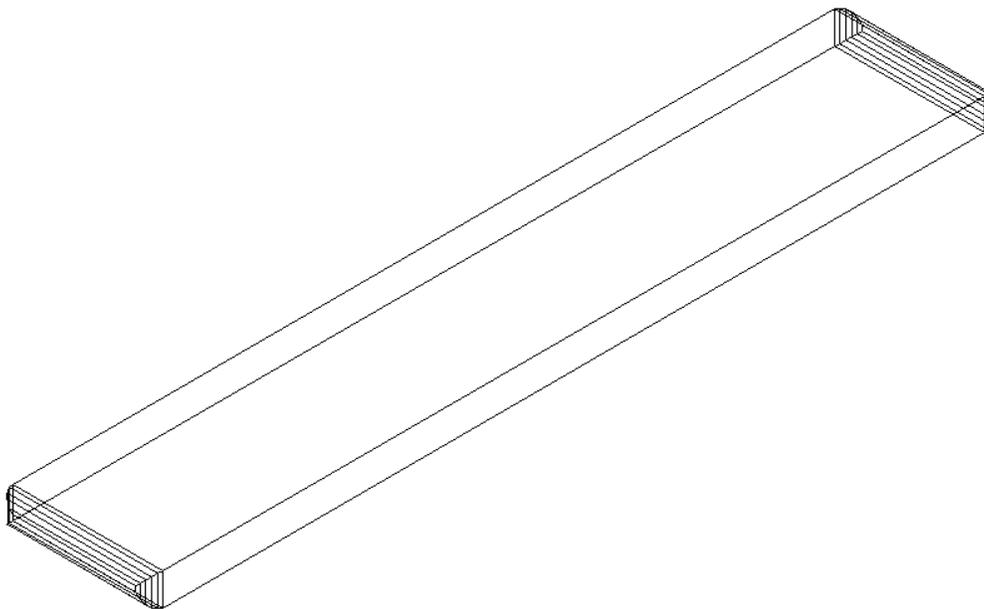


Figure 17.20 – The Finished Brace

To save Problem Three in your Foot Rest Assembly folder, select the Save tool on the Standard toolbar. The Save Drawing As window will appear in your graphical display. In the Save In list box, select your drawing folder. At the File Name textbox, type “Brace” and hit the Save button to save the drawing (Figure 17.21).

In larger projects like assemblies, you will want to save more often. In architectural floor plans and mechanical assemblies, plan on saving on a regular basis. Once you initially save the file, to repeat the process, choose the Save tool on the Standard toolbar. The Save As window will not appear since the file already has a name. So, save when you leave your desk. Save when you answer the phone. Save when you are in a discussion with someone. In every case, always save your file, whether distracted or the last save time exceeds 10 minutes.

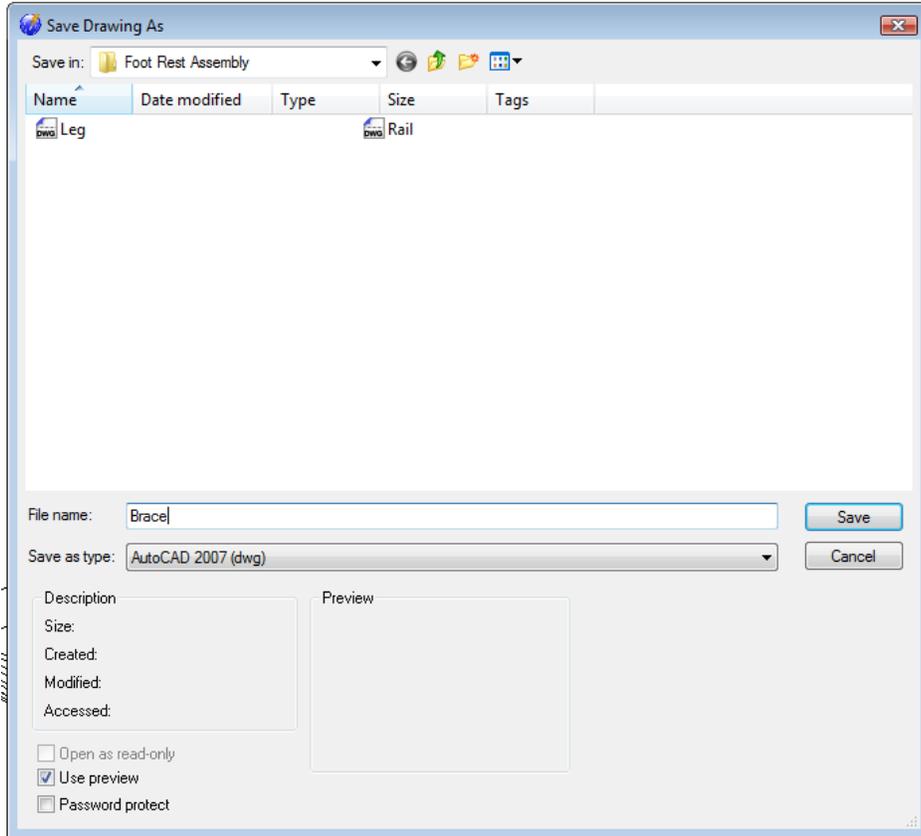


Figure 17.21 – Saving the Brace File in the Foot Rest Assembly Folder

Making and Saving Part Four, the Foot Rest

Part four is just a rectangular box with rounded ends on three sides. The dimensions of the geometric solid are 20.0 inches by 5.5 inches by 0.75 inches as shown in Figure 17.22. Again, the basic tools like the Box command will come in handy to this simple solid. You will add a fillet on two 0.75 corners. For this part you will learn to create a continuous fillet along three edges.

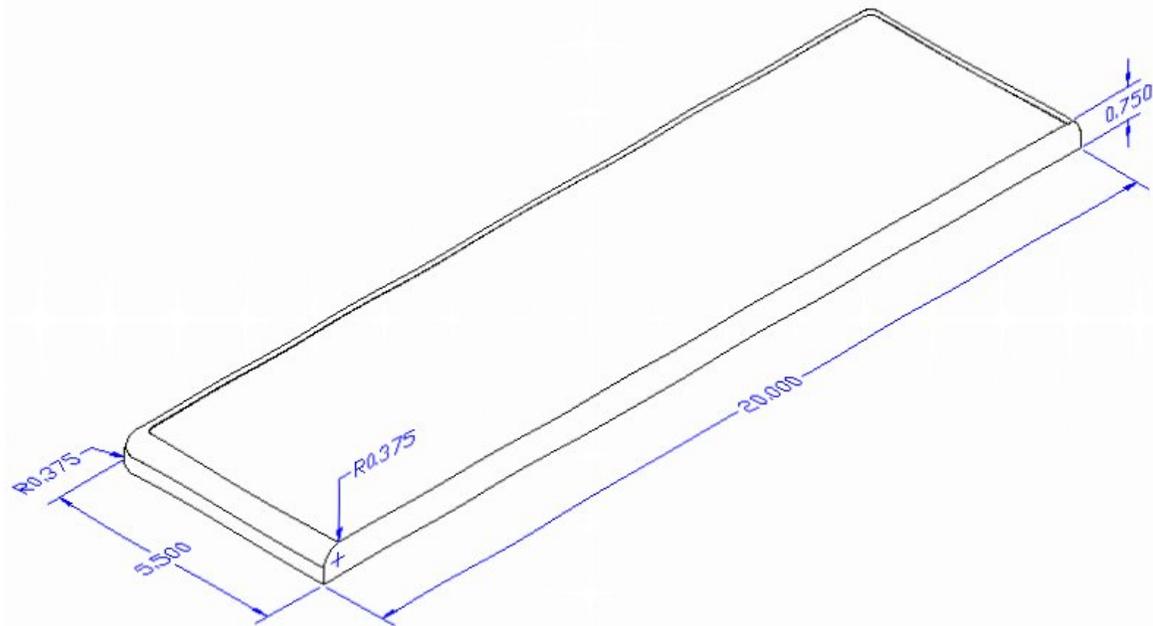
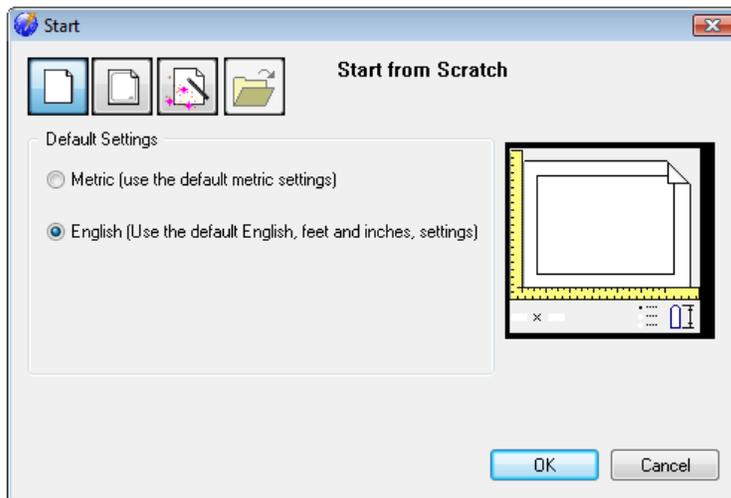


Figure 17.22 – Part Four Sketch



In Figure 17.23, the “Start” window is loaded by selecting the New tool on the Standard Toolbar. Select the “Start from Scratch” button at the top of the window. Select English as the default units setting and hit OK.



Figure 17.23 – Starting the Drawing Using the Mechanical Template

To draw the solid rectangular box, select the Box tool on the Solids toolbar. The command line prompts you to specify the corner or center of the box. Identify the corner of the box by picking anywhere in the middle of the graphical display. After picking the starting point for the box, again the system asks you to denote a corner; type **L** for length and hit **ENTER**. For this part, type **20.0** for the X coordinate, **5.5** for the y coordinate and **0.75** for the Z coordinate. Switch to the Southwest Isometric view so the solid appears in the graphical display as shown in Figure 17.24.

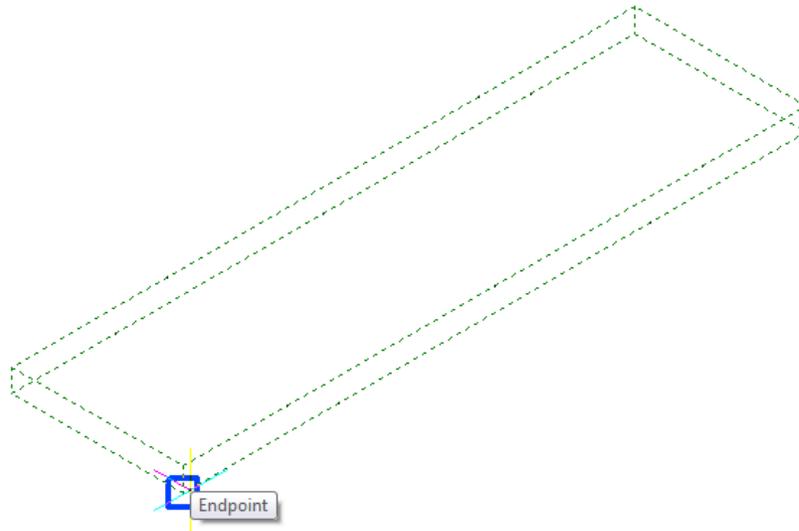


Figure 17.24 – Creating Part One and Moving the Part to the Origin

To move the part to the drawing origin of 0,0,0 on the x, y and z axis, pick the Move tool on the Modify toolbar, select the 3D solid, and hit ENTER. At the prompt to specify a base point, select the bottom corner closest to you, which will become the part's insertion point into the assembly drawing (Figure 17.24). Type **0,0,0** and hit **ENTER**.

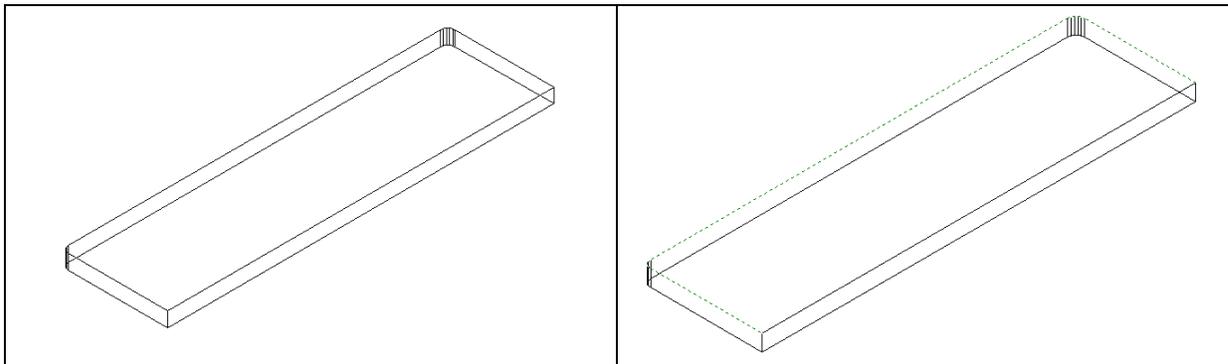


Figure 17.25 – Rounding the 0.75 Corner

Figure 17.26 – Selecting the Top Edge

Select the Fillet tool on the Modify toolbar. Type an “**R**” for radius and then **0.375** for the measurement. Select the vertical edge on the left side of the box and hit ENTER. Now select that edge again, as well as the other vertical edge on the left side of the part so that each line turns dotted and hit ENTER. The rounded edges will appear as shown in Figure 17.25.

Hit ENTER again to repeat the Fillet command. Select the top edge of the left face of the box and hit ENTER. Type **C** and ENTER to specify that you wish to use the Chain feature of the Fillet command. Now Select the back top edge of the part; that edge as well as the farthest top edge will become dotted (Figure 17.26). Hit ENTER to complete the fillet (Figure 17.27).

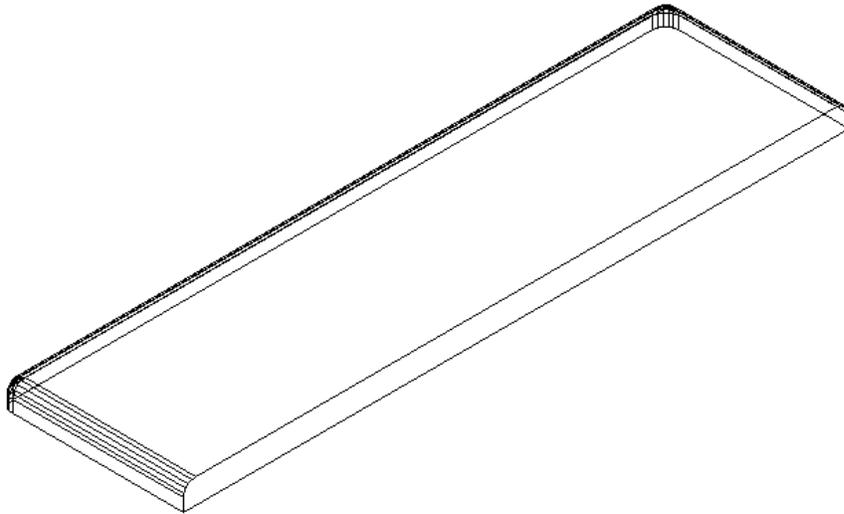


Figure 17.27 – Saving the completed Fillet chain

To save Part Four as shown in Figure 17.27 in your Foot Rest Assembly folder, select the Save tool on the Standard toolbar. The Save Drawing As window will appear in your graphical display. In the Save In list box, select your drawing folder. At the File Name textbox, type “Foot Rest” and hit the Save button to save the drawing (Figure 17.28).

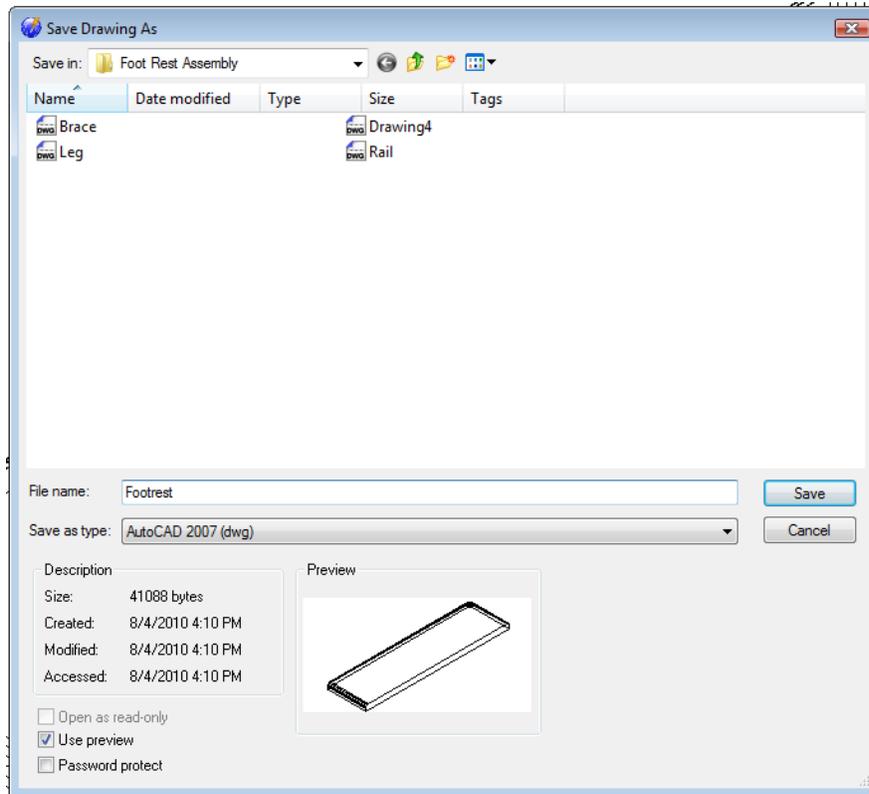


Figure 17.28 – Saving the Foot Rest File in the Foot Rest Assembly Folder

Making an Assembly Drawing

You will make the Foot Rest Assembly drawing from the four solids parts in your Foot Rest Assembly Folder. Following the sketch as shown in Figure 17.29 and using the Endpoint and the Midpoint Esnaps, you will easily construct your product.

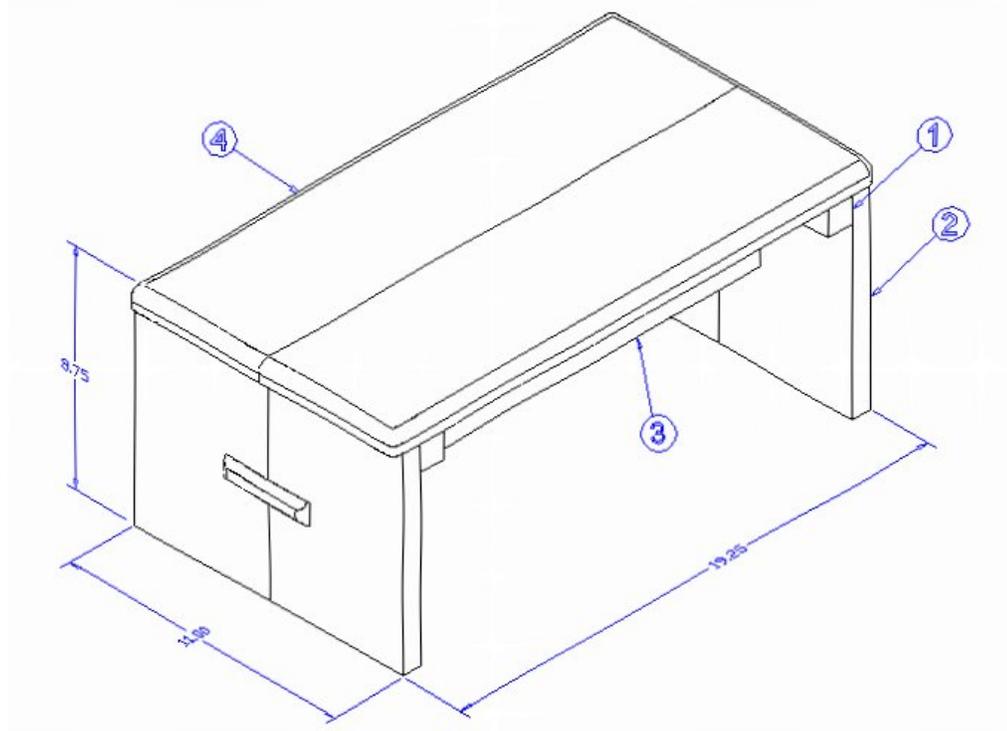
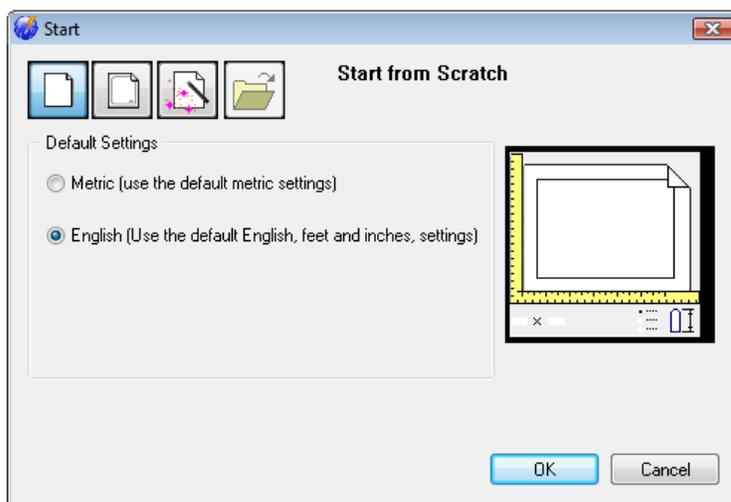


Figure 17.29 – The Foot Rest Assembly Sketch



In Figure 17.30, the “Start” window is loaded by selecting the New tool on the Standard Toolbar. Select the “Start from Scratch” button at the top of the window. Select English as the default units setting and hit OK.



Figure 17.30 – Starting the Drawing Using the Mechanical Template

To save the Foot Rest Assembly in your Foot Rest Assembly folder, select the Save tool on the Standard toolbar. The Save Drawing As window will appear in your graphical display. In the Save In list box, select your drawing folder. At the File Name textbox, type “Foot Rest Assembly” and hit the Save button to save the drawing (Figure 17.31).

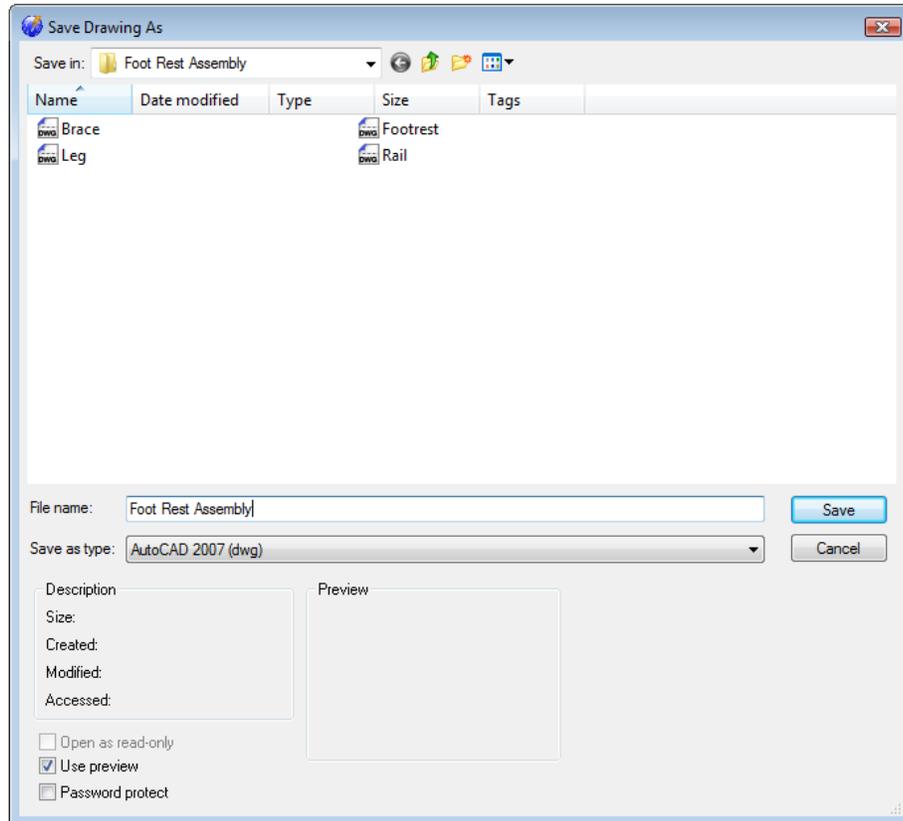


Figure 17.31 – Saving the Foot Rest Assembly File in the Foot Rest Assembly Folder

Using the XREF Tool to Import the First Part Drawing

Now you have finished all of the hard work to make the product by creating each individual solid and saving them into the Foot Rest Assembly folder. Inserting a reference to the solid component is as simple as inserting a block or external file. On the View toolbar, select the Southwest Isometric View tool to allow you to see the part on the XY plane and the vertical Z-axis will rise from the level surface.



Select Xref Manager tool on the Reference toolbar and the Xref Manager Window will appear in the center of the graphical display. Presently, the Foot Rest Assembly drawing does not show any attached files as shown in Figure 17.32. To attach an externally referenced file, click on the Attach button and the Select File to Attach window will open. Keeping the Reference Type as Attachment and the other options as shown in Figure 17.33, hit OK and place the leg into the drawing.

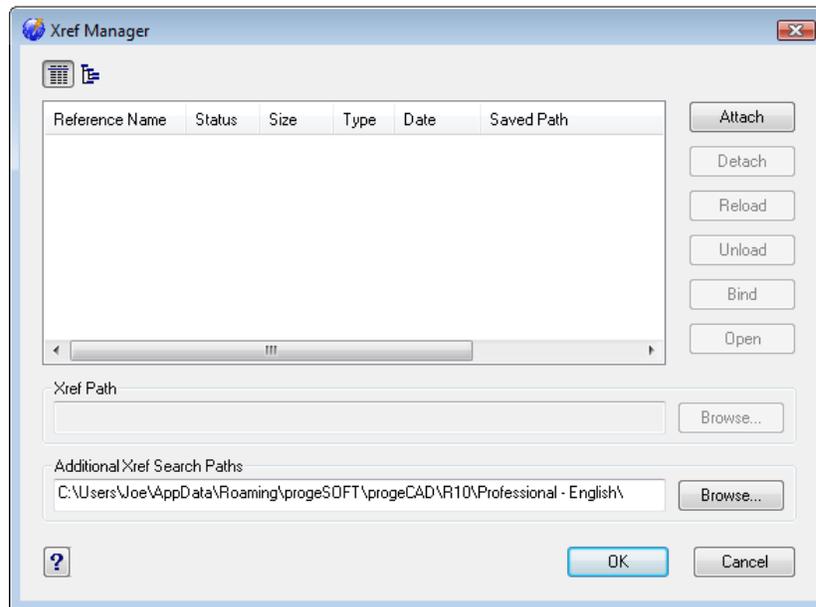


Figure 17.32 – The Xref Manager Window

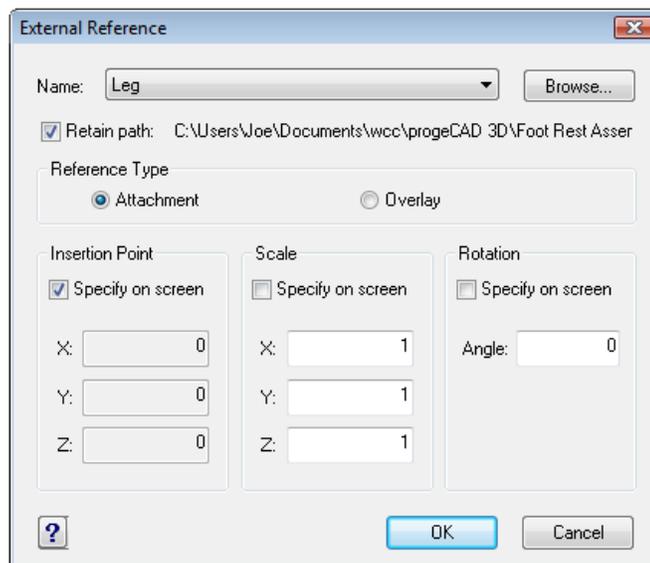


Figure 17.33 – Selecting the Leg in the External Reference Window

The Xref Manager will reappear, now listing the Leg as External Reference. Hit OK to exit the Xref Manager. Zoom Extents to see the part in full view.

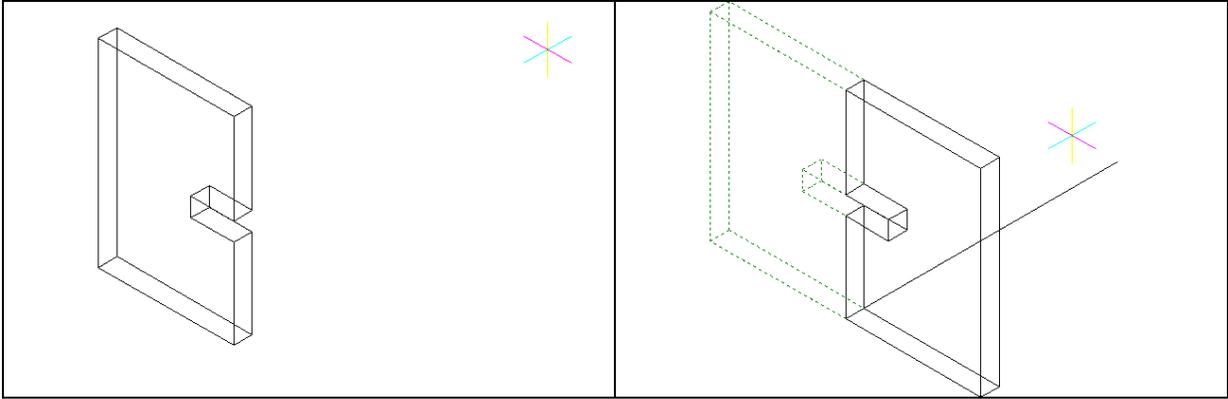


Figure 17.34 – Inserting the Leg

Figure 17.35 – Mirroring the Leg

You will use the Mirror tool on the Modify toolbar to copy the Leg across itself. At the prompt to select entities to mirror, pick the leg and then hit ENTER to proceed to the second half of the Mirror command. Place the “Start of mirror line:” by picking the right Endpoint as shown in Figure 17.35 and the “End of mirror line:” by picking to the right of the part with your Ortho “On”. Hit **ENTER** to keep the original leg and the mirrored and duplicated entity from the top half will appear on the bottom right of the graphical display.

Using the XREF Tool to Import the Second Part Drawing

Pick the External Reference tool on the Reference toolbar and the Select File to Attach window will appear in the center of the display.

The second externally referenced drawing file that you will want to insert into the assembly drawing is the brace, so select the brace file from the list of drawings in the Foot Rest Assembly folder (Figure 17.36). Pick the Open button to close the dialog box, and then place the brace anywhere to the right of the legs. Hit ENTER to approve the 1.0 scale value on the x and y axis, as well as to keep the rotation angle at 0.

To move the Brace, select the Move tool on the Modify toolbar, select the 3D solid, and hit ENTER. At the prompt to specify a base point, select the bottom corner closest to you, where the rounded edge meets the flat face of the bottom corner. The point of displacement is the Endpoint on the lowest point of the slot as shown in Figure 17.38.

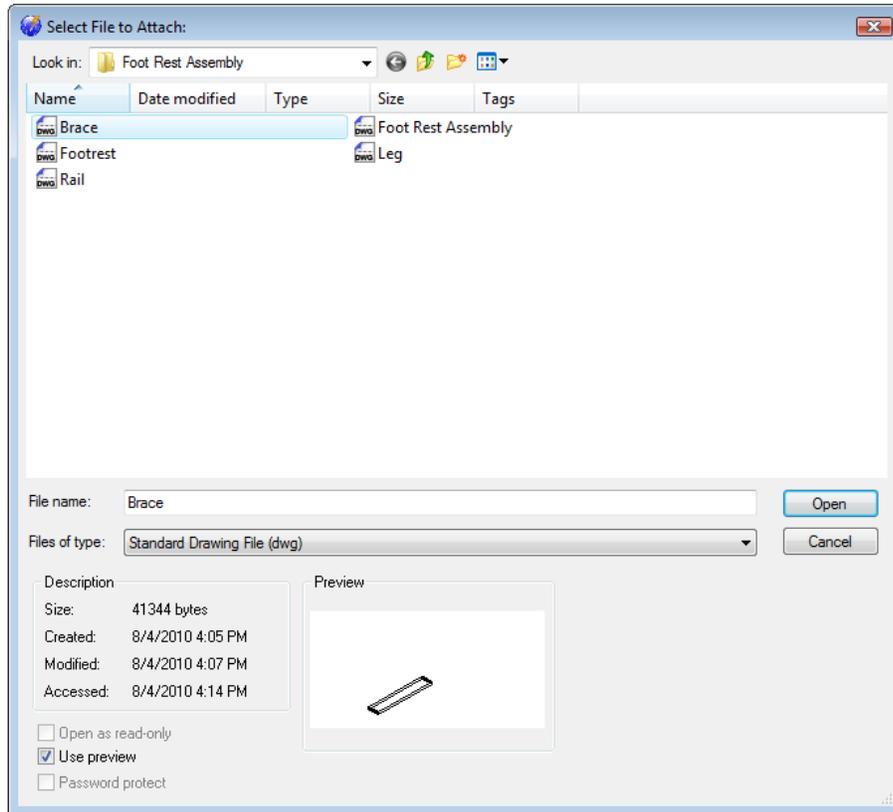


Figure 17.36 – Selecting the Brace File in the Select Reference File Window

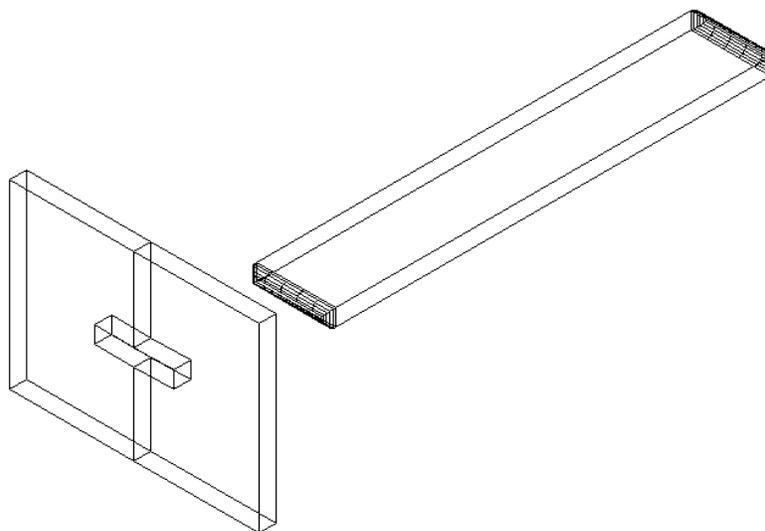


Figure 17.37 – Inserting the Brace in the Foot Rest Assembly

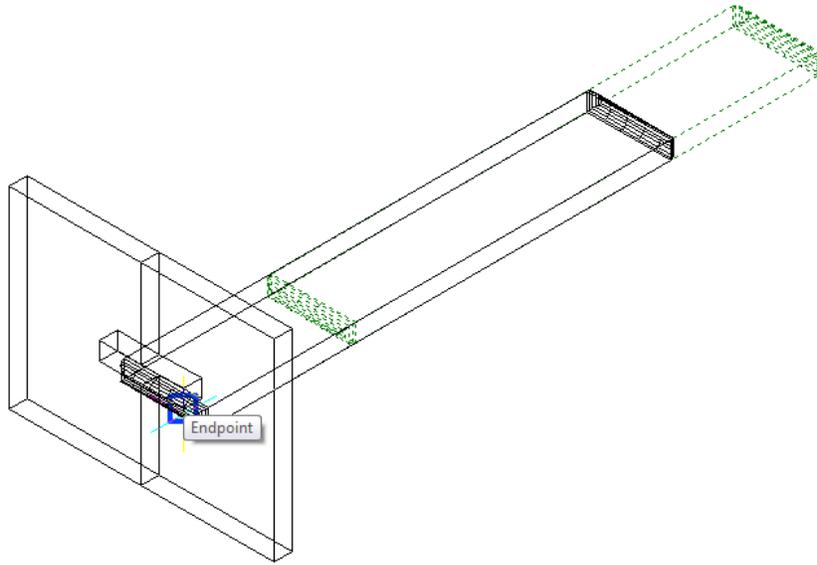


Figure 17.38 – Moving the Brace into Position in the Slot in the Leg

The Brace will move into the slot with a perfect fit. The curved end will stick out of the rectangular hole and you can check your work by changing to the Front View on the View toolbar. If the Brace is in the correct position, return to your previous view, and if the Brace is not located properly, use the Move command to reposition the solid. The use of Entity Snaps in the creation of assembly layouts proves just as valuable as it did in individual part drawings.

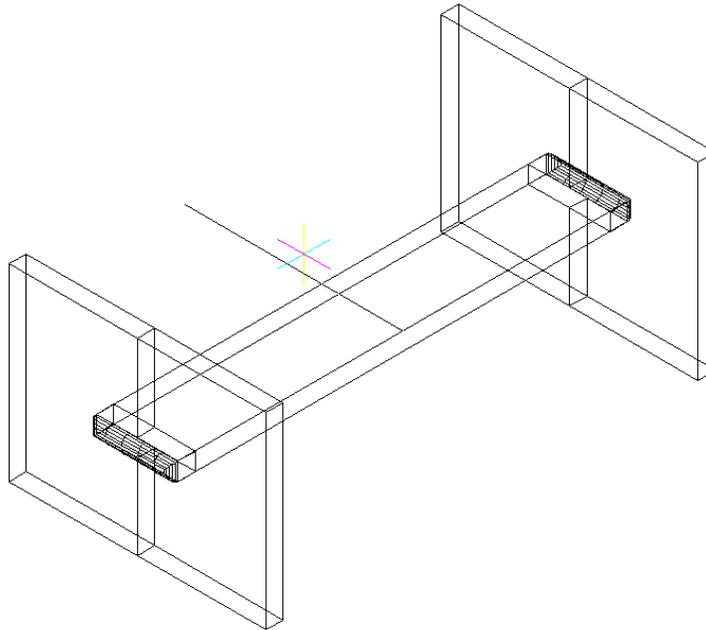


Figure 17.39 – Mirroring the Leg to the Opposite Side of the Brace

You will use the Mirror tool on the Modify toolbar to copy the Leg across the Midpoint of the Brace. At the prompt to select objects, pick the two Legs and then hit **ENTER** to proceed to the second half of the Mirror command. Specify the “Start of mirror line:” by picking the brace’s Midpoint as shown in Figure 17.39. Specify the “End of mirror line:” by moving your cursor with your Ortho “On” until the preview shows two legs on the empty side of the brace. The default to delete the source objects is no, so at the prompt to delete the first two legs just hit **ENTER** and the mirrored and duplicated entities from the left side will appear on the top and a right of the graphical display.

Using the XREF Tool to Import the Third Part Drawing

Pick the External Reference tool on the Reference toolbar and the Select File to Attach window will appear in the center of the display.

The third externally referenced drawing file that you will want to insert into the assembly drawing is the rail, so select the rail file from the list of drawings in the Foot Rest Assembly folder. Pick the Open button to close the dialog box, and then place the rail anywhere to the right of the legs. Hit ENTER to approve the 1.0 scale value on the x and y axis, as well as to keep the rotation angle at 0.

To move the rail, select the Move tool on the Modify toolbar, select the 3D solid, and hit ENTER. At the prompt to specify a base point, select the upper Endpoint of the edge closest to you on the rail. The point of displacement is the Endpoint on the top right Endpoint of the Leg as shown in Figure 17.42.

You will use the Mirror tool on the Modify toolbar to copy the rail across the Midpoint of the Brace. At the prompt to select objects, pick the rail and then hit **ENTER** to proceed to the second half of the Mirror command. Specify the “Start of mirror line:” by picking the brace’s Midpoint. Specify the “End of mirror line:” by moving your cursor with your Ortho “On” until the preview shows a rail on the other side of the brace as shown in Figure 17.43. The default to delete the source objects is no, so at the prompt to delete the first rail just hit **ENTER** and the mirrored and duplicated entity from the left side will appear on the top and a right of the graphical display.

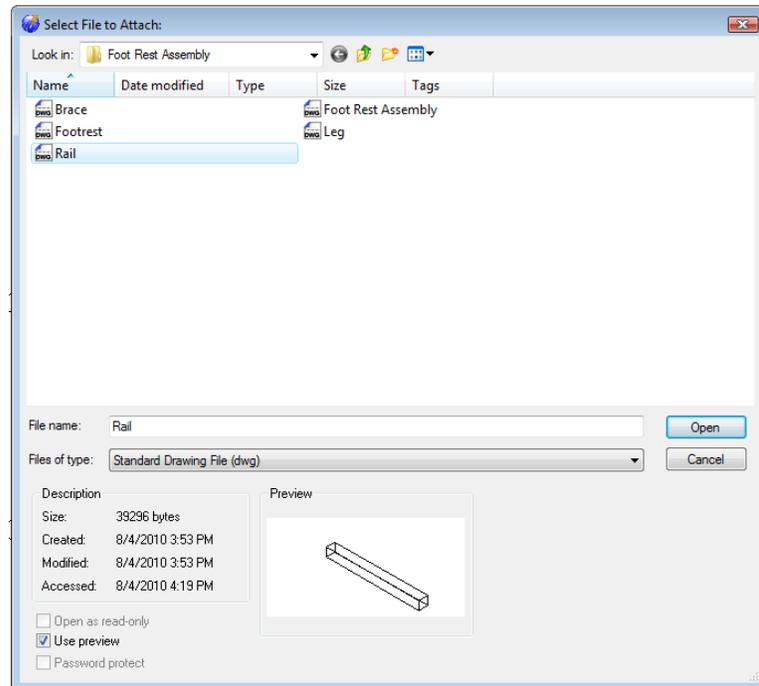


Figure 17.40 – Selecting the Rail File in the Select Reference File Window

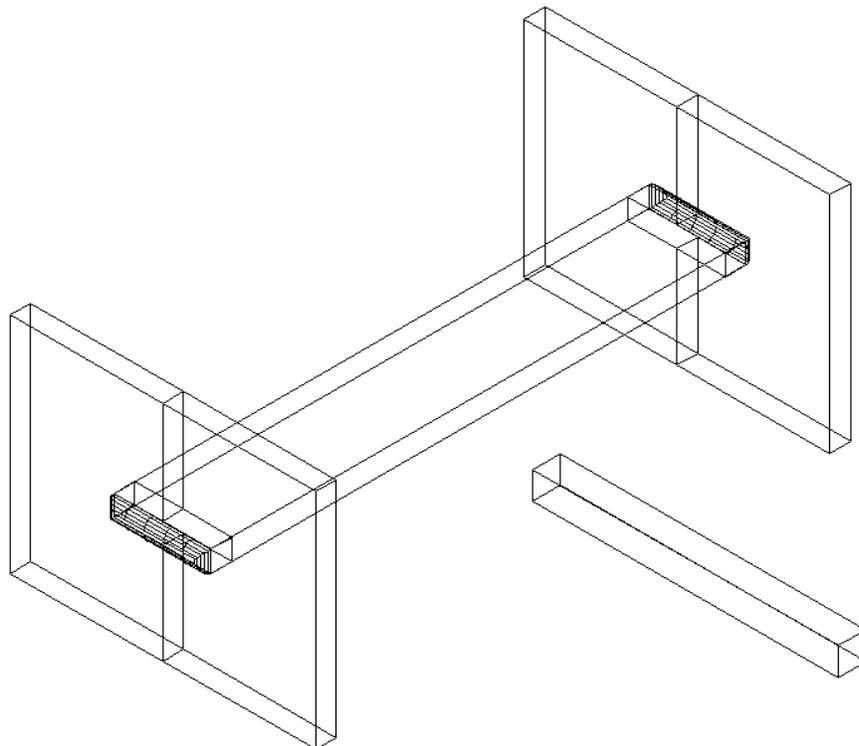


Figure 17.41 – Inserting the Rail in the Foot Rest Assembly Drawing

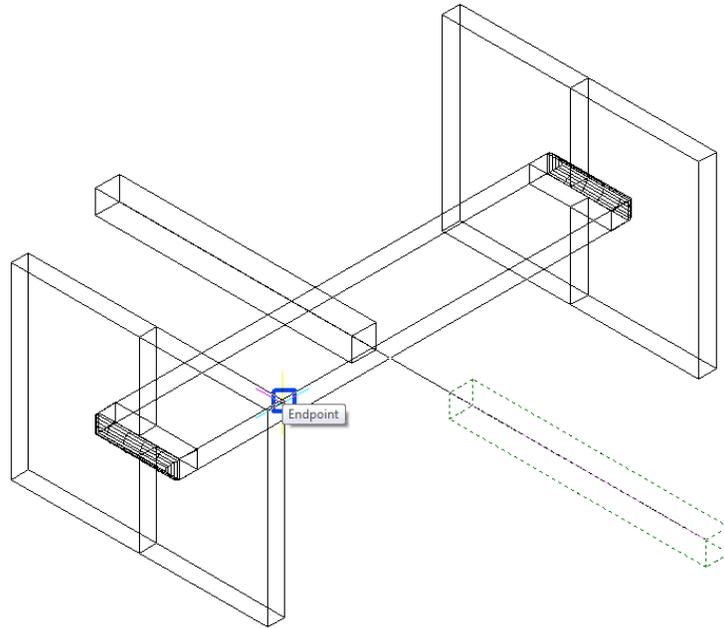


Figure 17.42 – Moving the Rail in the Foot Rest Assembly Drawing

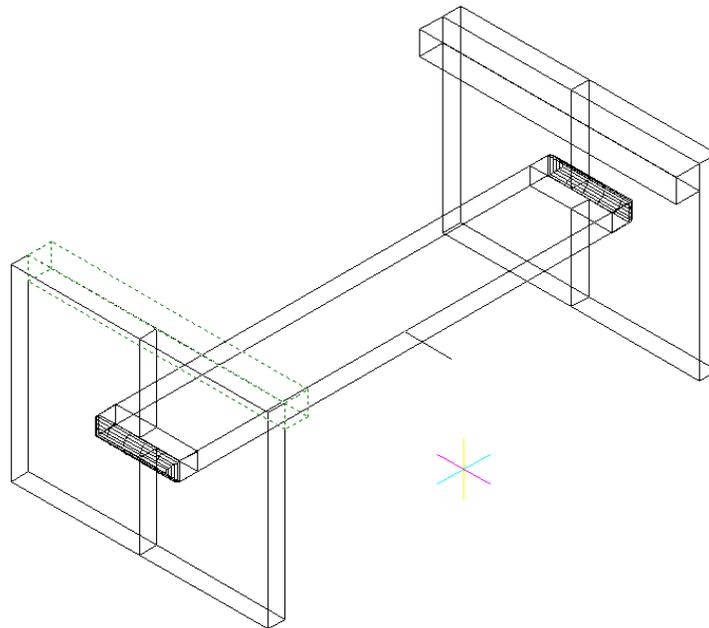


Figure 17.43 – Mirroring the Rail in the Foot Rest Assembly Drawing

Using the XREF Tool to Import the Fourth Part Drawing

Pick the External Reference tool on the Reference toolbar and the Select File to Attach window will appear in the center of the display.

The fourth externally referenced drawing file that you will want to insert into the assembly drawing is the footrest, so select the footrest file from the list of drawings in the Foot Rest Assembly folder. Pick the Open button to close the dialog box, and then place the footrest on an Endpoint on the outside joint of the Leg as shown in figure 17.45. Hit ENTER to approve the 1.0 scale value on the x and y axis, as well as to keep the rotation angle at 0.

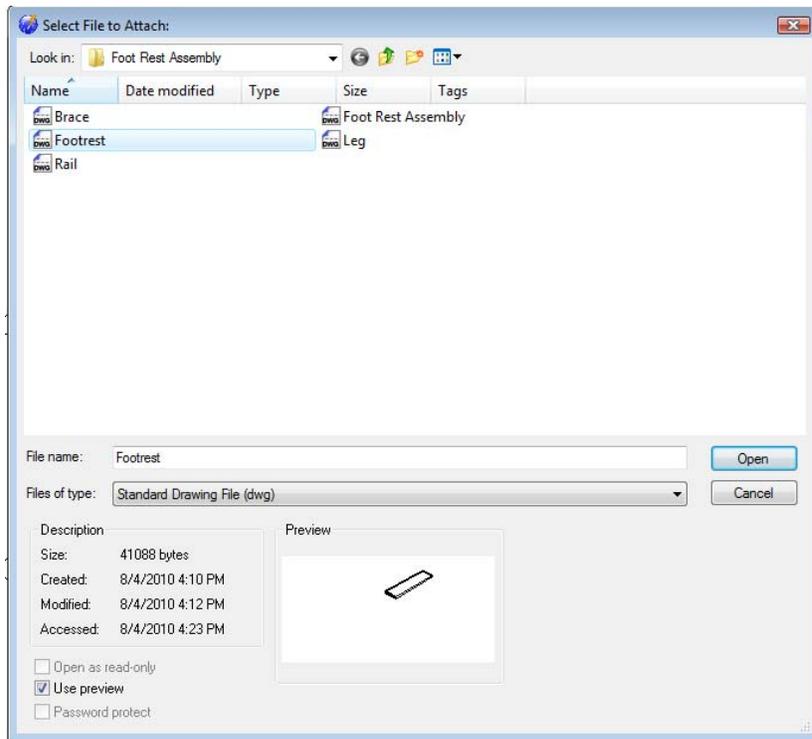


Figure 17.44 – The Xref Manager Window Showing the Attached Files

To move the foot rest, select the Move tool on the Modify toolbar, select the 3D solid, and hit ENTER. At the prompt to specify a base point, select anywhere in the graphical display. Move your mouse to the left and type **0.375** and **ENTER** to move the footrest into the proper position (Figure 17.46).

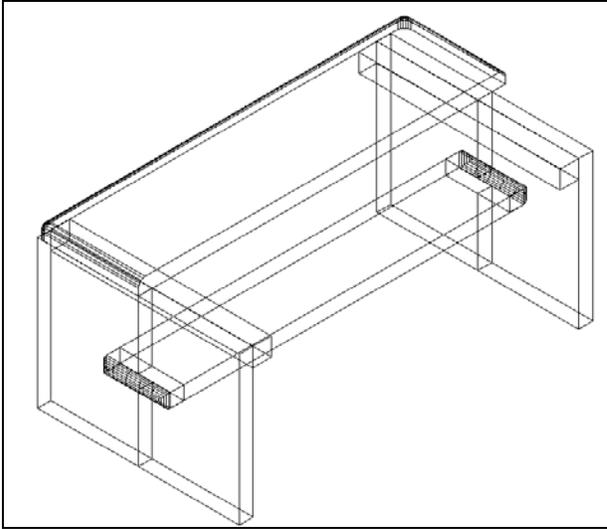


Figure 17.45 – Inserting the Foot Rest

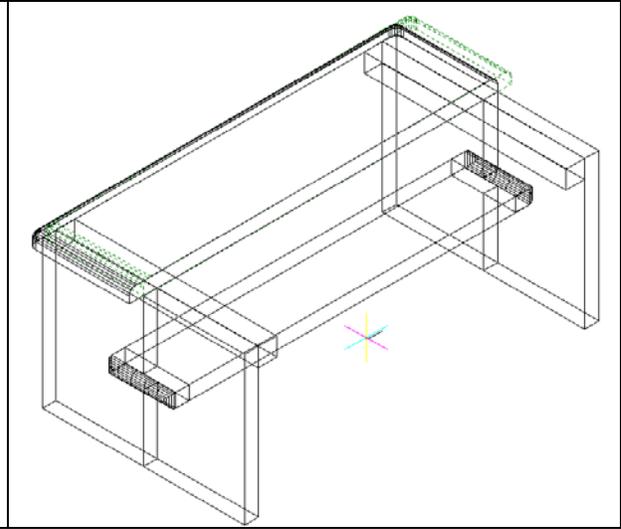


Figure 17.46 – Moving the Foot Rest

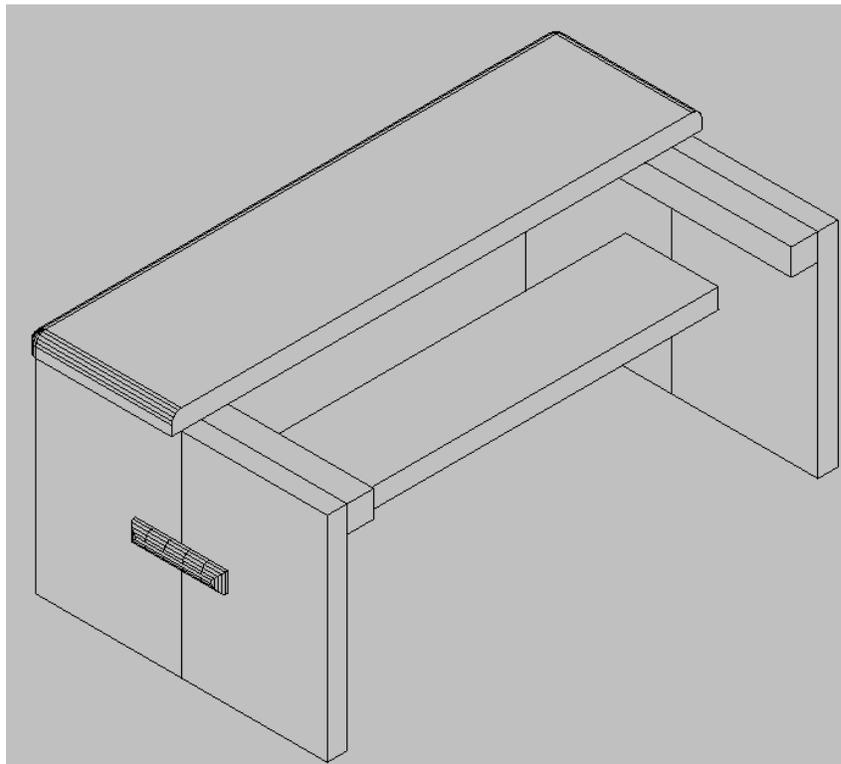


Figure 17.47 – Using the Hide Command to View the Assembly

You will use the Mirror tool on the Modify toolbar to copy the Footrest across the Endpoint of the Leg Joint. At the prompt to select objects, pick the footrest and then hit **ENTER** to proceed to the second half of the Mirror command. Specify the “Start of mirror line:” by picking the Endpoint as shown in Figure 17.48. Specify the “End of mirror line:” by picking to the lower left of the part with your Ortho “On”. The default to delete the source objects is no, so at the prompt to delete the first rail just hit **ENTER** and the mirrored and duplicated foot rest from the left side will appear on the right side of the assembly as well.

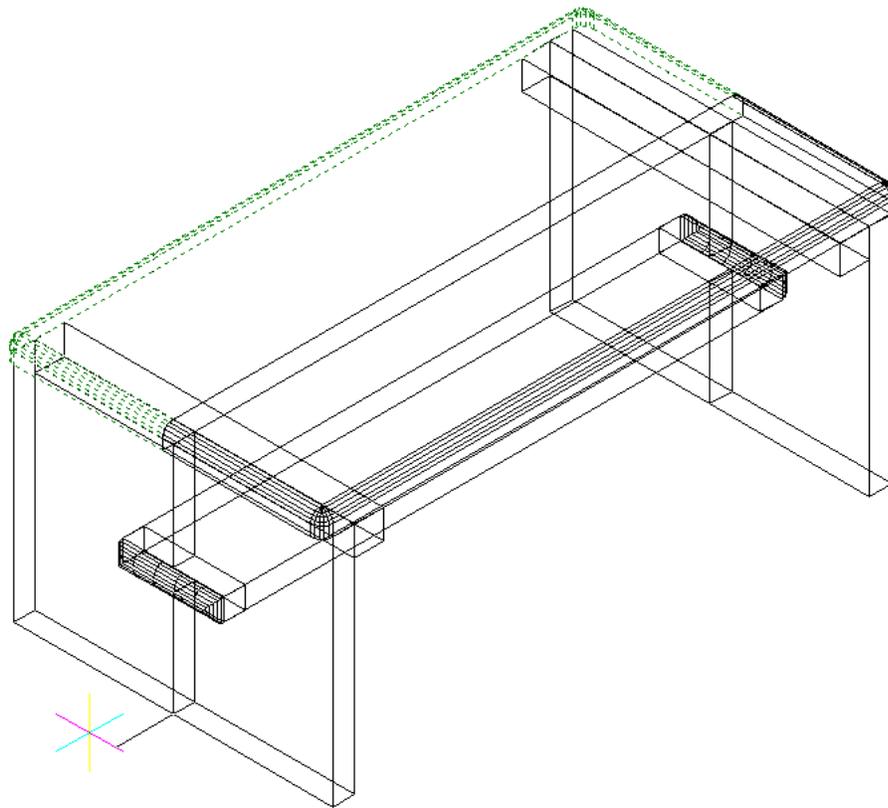


Figure 17.48 – Mirroring the Foot Rest in the Foot Rest Assembly Drawing

To save the Foot Rest Assembly in your Foot Rest Assembly folder, select the Save tool on the Standard toolbar. The current and assembled version is now updated in the Foot Rest Assembly file. Your drawing should appear as shown in Figure 17.49.

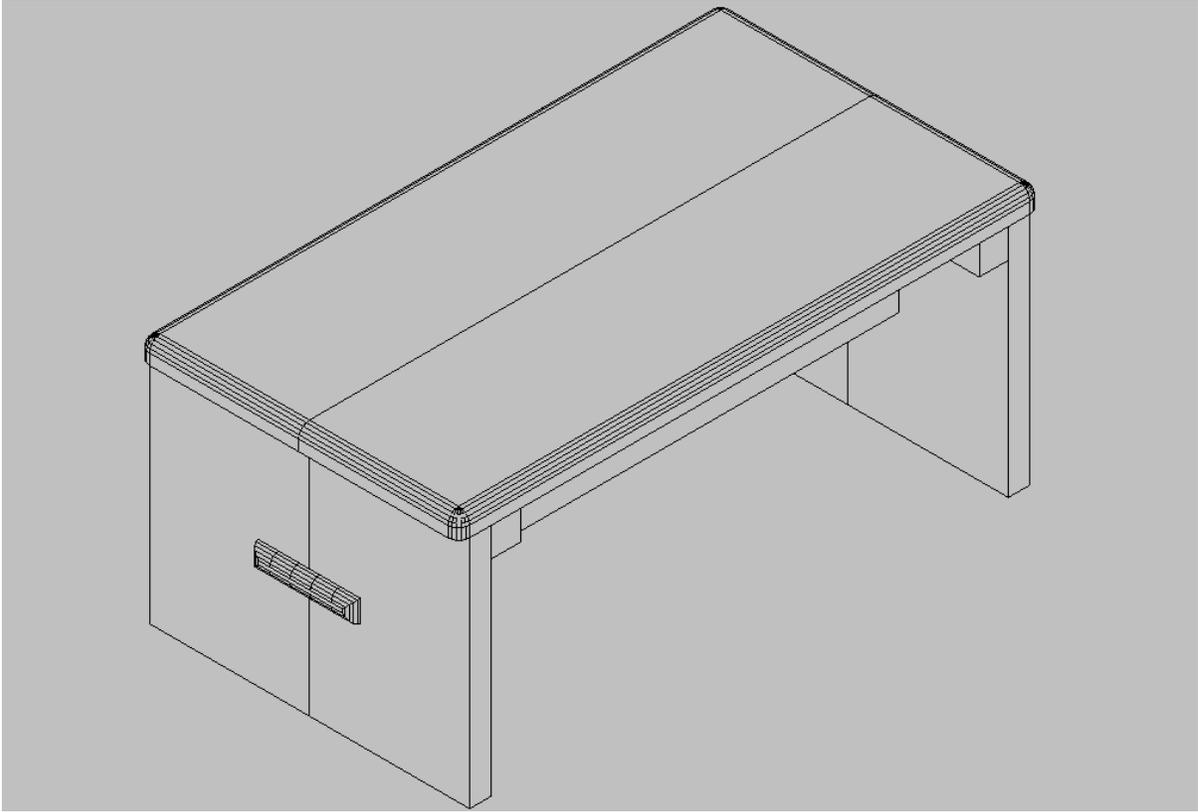


Figure 17.49 – The Finished Foot Rest Assembly Drawing

*** World Class CAD Challenge 101-27* - Close this drawing file. Create four solid drawings in an assembly folder, and make an assembly file from the four externally referenced files. Complete the task in less than 60 minutes. Continue this drill four times, each time completing the drawing under 60 minutes to maintain your World Class ranking.**

*** World Class CAD Challenge * - Report your best times to World Class CAD at www.worldclasscad.com to obtain your world class ranking.**